

3D PRINTER INDUCTION - THE EDGE



State Library
of Queensland

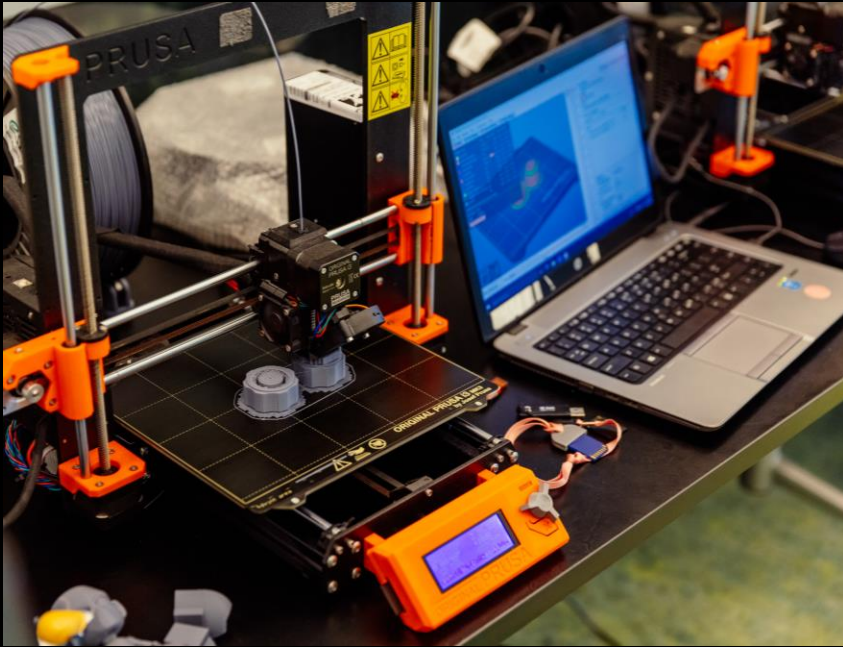


Queensland
Government



Acknowledgement of Country

We acknowledge Aboriginal and Torres Strait Islander peoples and their continuing connection to land and as custodians of stories for millennia. We respectfully acknowledge the land on which we all meet today, and pay our respects to elders past, present and emerging.



INDUCTION SUMMARY

Using the State Library 3D printers, you can turn a digital 3D model into a plastic object.

In this induction you will learn the basics of 3D modelling with Tinkercad and how to print your own model, including how to load it into the printer and cleaning up the product.

Once you've completed the induction you can book and use the State Library of Queensland's 3D Printers at The Edge during Open lab.

Open Lab sessions are a chance to meet up with like-minded makers and tinkerers at The Edge with facilitators to support your creative needs in the space.

Bookings are required to use the equipment and you will be able to book with your SLQ account once you have completed the relevant induction.

Wednesday 1.30pm – 8pm

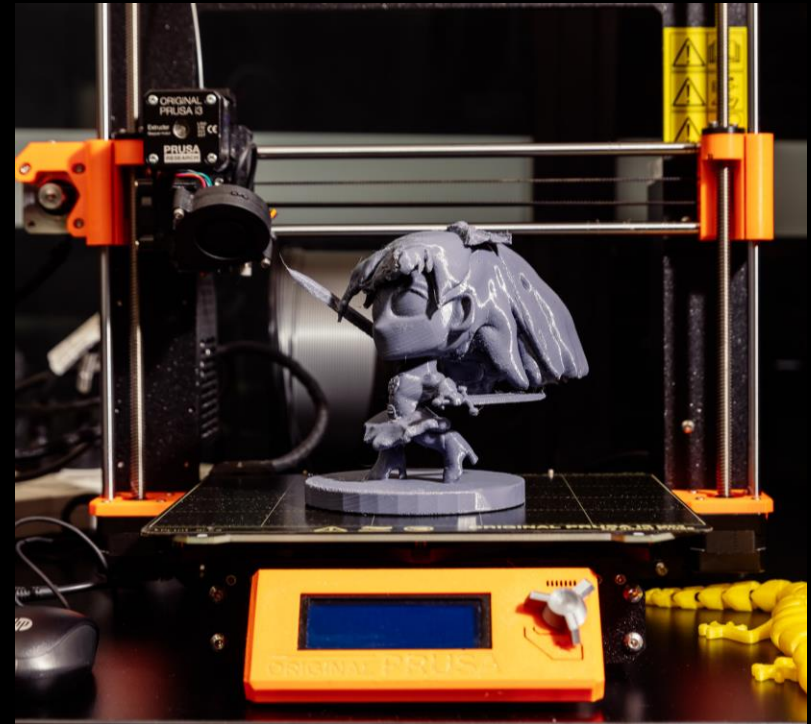
Thursday 1.30pm – 8pm

Saturday 12pm – 6pm

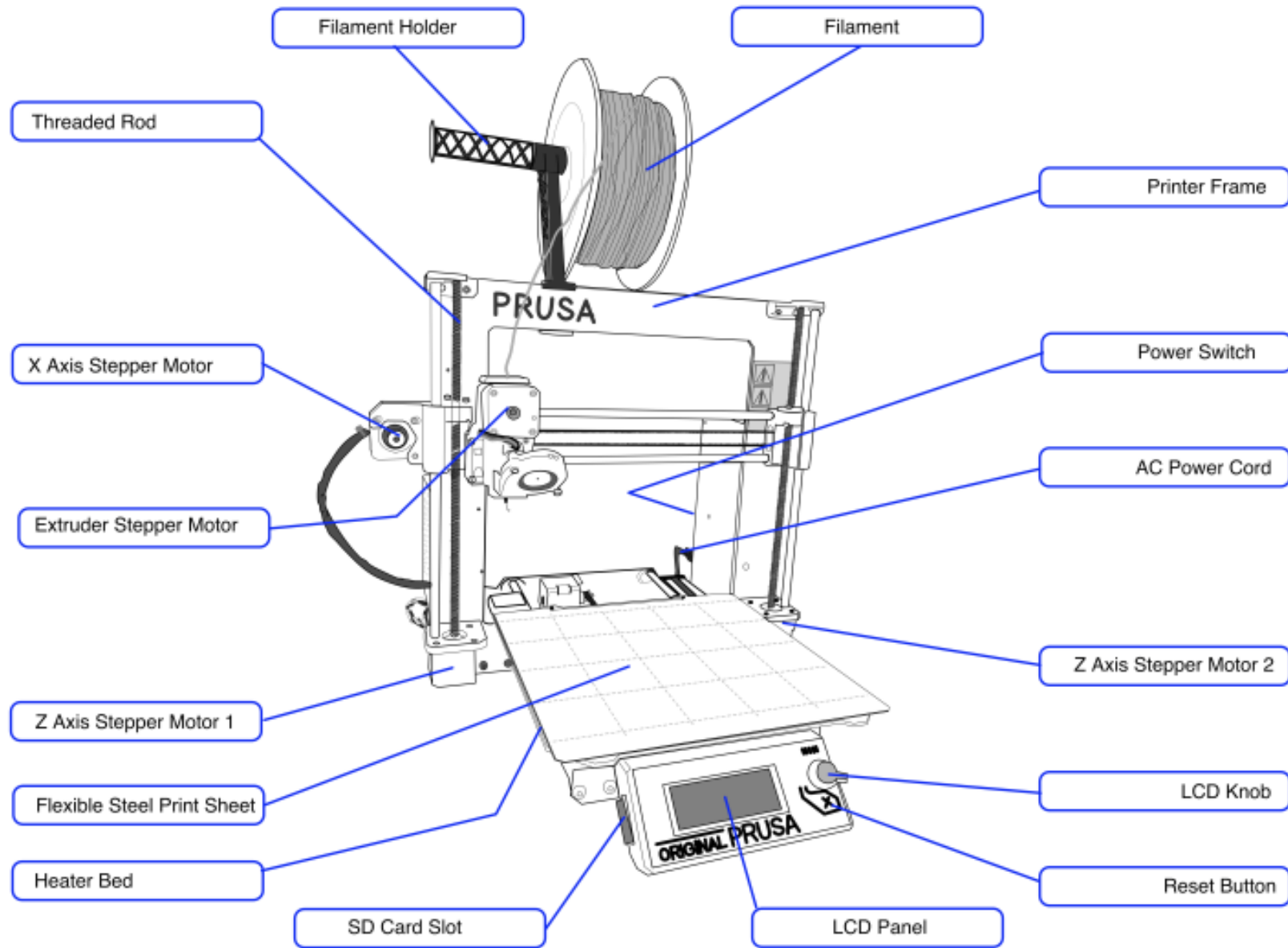
For more info and to book, head to <https://www.slq.qld.gov.au/visit/spaces/edge>

What is a 3D PRINTER?

- Allows for 3D printing which is the construction of a three-dimensional object. It does this through material being deposited, joined or solidified under computer control.
- The Edge has 5 Prusa model i3 MK3s+ printers, which are the successor to Original Prusa i3 MK3 and feature hardware and software upgrades offering improved reliability and ease of use.



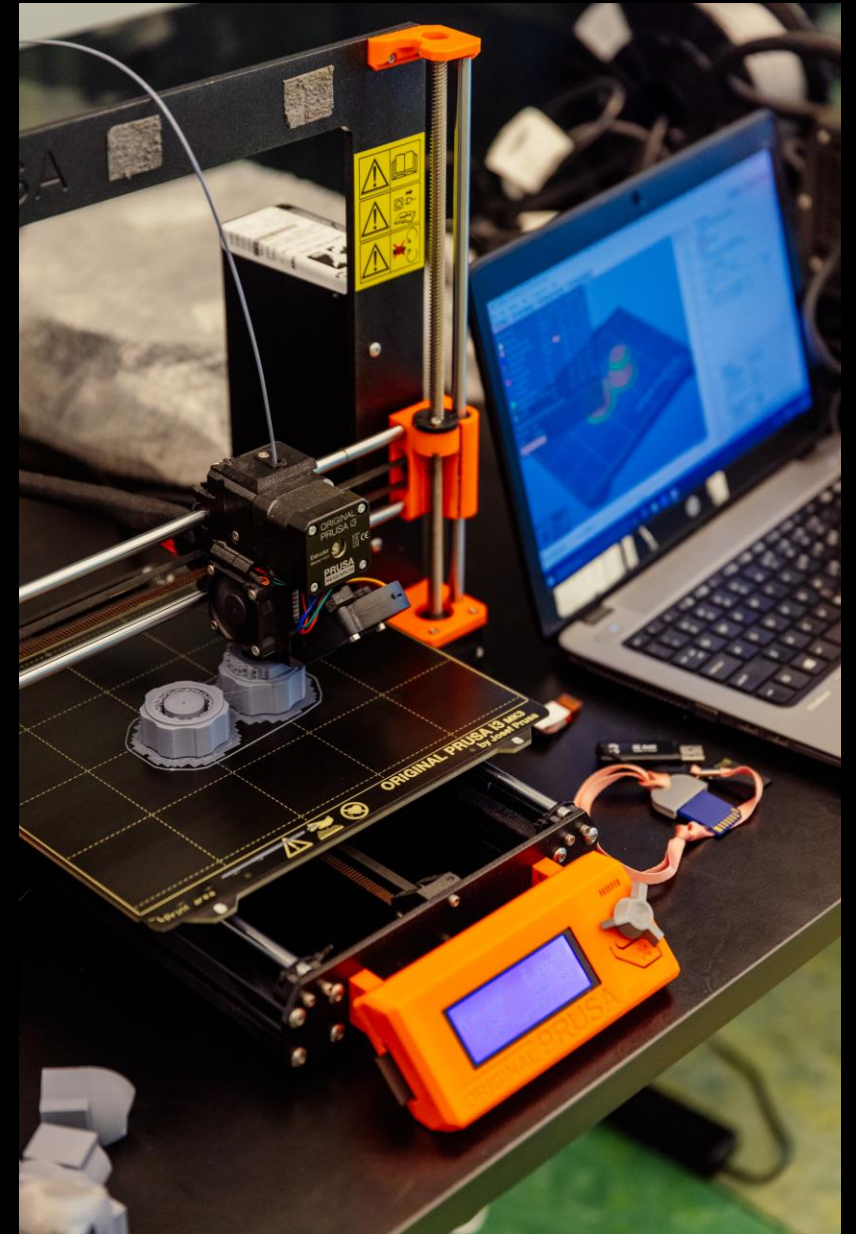
EQUIPMENT OVERVIEW



MACHINE OVERVIEW

Getting Started

- Make a 3D printer booking.
- Set up design file or if already done, import your model into PrusaSlicer.
- Staff members can make recommendations regarding part orientation, support placement, and layer height.
- When ready to print grab a 3D printer form and ask an Open lab staff member to approve your job!



Safety

- Electricity
- Fire
- Respiratory
- Crush injury
- Isopropyl
(eyes/breathing/flammable)

Check the safe operating procedures regularly and ask a staff member for help if you have any problems or concerns.

SL **SAFE OPERATING PROCEDURE**
all yours
Prusa i3 MK3S+ 3D printer

DO NOT use this machine unless you have completed an induction and the Supervisor has given permission

Cut-resistant gloves must be worn when removing support structure from 3D prints.

Protective eyewear must be worn when removing support structure from 3D prints.

Fume filter serving printer enclosure must be operating when print job is underway.

Printer hot end is a potential source of combustion. Keep isopropyl alcohol at least 2 metres away.

PRE-OPERATIONAL SAFETY CHECKS

1. Only filament approved by the Fab Lab is permitted for use on the Prusa printers.
2. Print job G-code files must be prepared using PrusaSlicer software installed on Fab Lab computers
3. ONLY to ensure correct printer and filament settings. The default file name proposed when exporting G-code includes critical information required by the Supervisor. If editing the file name, only edit text appearing prior to the 1st underscore.
4. Sliced models must be assessed by Supervisor prior to saving print job G-code.
5. Remove the steel heatbed sheet to the cleaning station for wiping with isopropyl alcohol prior to starting a print job; DO NOT take the isopropanol or wetted towels to the printer.
6. Smooth steel heatbed sheets are to be used for printing with PLA filament, and textured sheets for use with PETG. Smooth sheets MUST NOT be used when printing PETG filament.
7. All printers include sheet profiles for PLA and PETG. Ensure the correct profile is selected in the settings for the sheet being used.
8. The ultrafine particles and VOCs generated when melting plastic filament pose a health hazard. The fume filter must be switched on, both extraction and return air gates opened, and the print enclosure door shut before preheating the printer or commencing a print job.
9. Faulty equipment must not be used. Immediately report any equipment concerns to the Supervisor.

OPERATIONAL SAFETY CHECKS

1. Upon inserting an SD card into the printer slot and selecting a file to print, the nozzle and print bed will commence heating. Once operational temperatures are reached, the print head and bed will begin moving. This movement presents potential pinch points. Keep fingers clear during operation.
2. Monitor the first printed layer to verify the filament has attached to the steel sheet properly.
3. In the event of a print failure press the Reset button. This will abort the job and force a full reset of the printer.
4. Remove the heatbed sheet from printer before attempting to remove a printed part from sheet. Wait for the sheet to cool, then flex the sheet. The print should pop off. DO NOT use metal tools to prise a print from the sheet.
5. Always wear safety glasses and protective gloves when removing support materials

HOUSEKEEPING

1. Ensure all printed material is removed from the heatbed sheet.
2. Switch off fume extractor when print job is complete.
3. Dispose of all support structure and other print waste in the appropriate bin.

POTENTIAL HAZARDS

- Hot surfaces and materials
- Sharp edges
- Hazardous emissions
- Molten plastics
- Eye injury
- Control errors
- Pinch points
- 240v electrical components
- Failure or malfunction

The Edge Fabrication Lab Safe Operating Procedure
Authorised by Daniel Flood
Uncontrolled when printed

State Library of Queensland
Issue Date: 14/07/2020
Version # 1

Records File # 520_315_227
Revision Date: 01/09/2023
1 | Page

Designing for 3D Printer

Fusion 360

Pros

- Powerful professional CAD, CAM, CAE software
- Lots of good tutorials online
- Great for designing close tolerance mechanical parts
- Free hobbyist licence available

Cons

- A lot to get your head around when starting out
- Full version relatively expensive (but not as expensive as others)

Designing for 3D Printer

Meshmixer

Pros

- Free
- Great software for sculpting 3D
- Good for repairing models made in other programs

Cons

- Probably want to try something more intuitive first to get your head around the concepts

Designing for 3D Printer

Blender

Pros

- Free and open source
- Great software for all types of 3D modelling and animation
- Lots of tutorials and guides online

Cons

- Probably want to try a simpler software first to get your head around the concept of 3D design

Designing for 3D Printer

TinkerCAD

Pros

- Free
- Great intuitive software for getting started
- Browser based

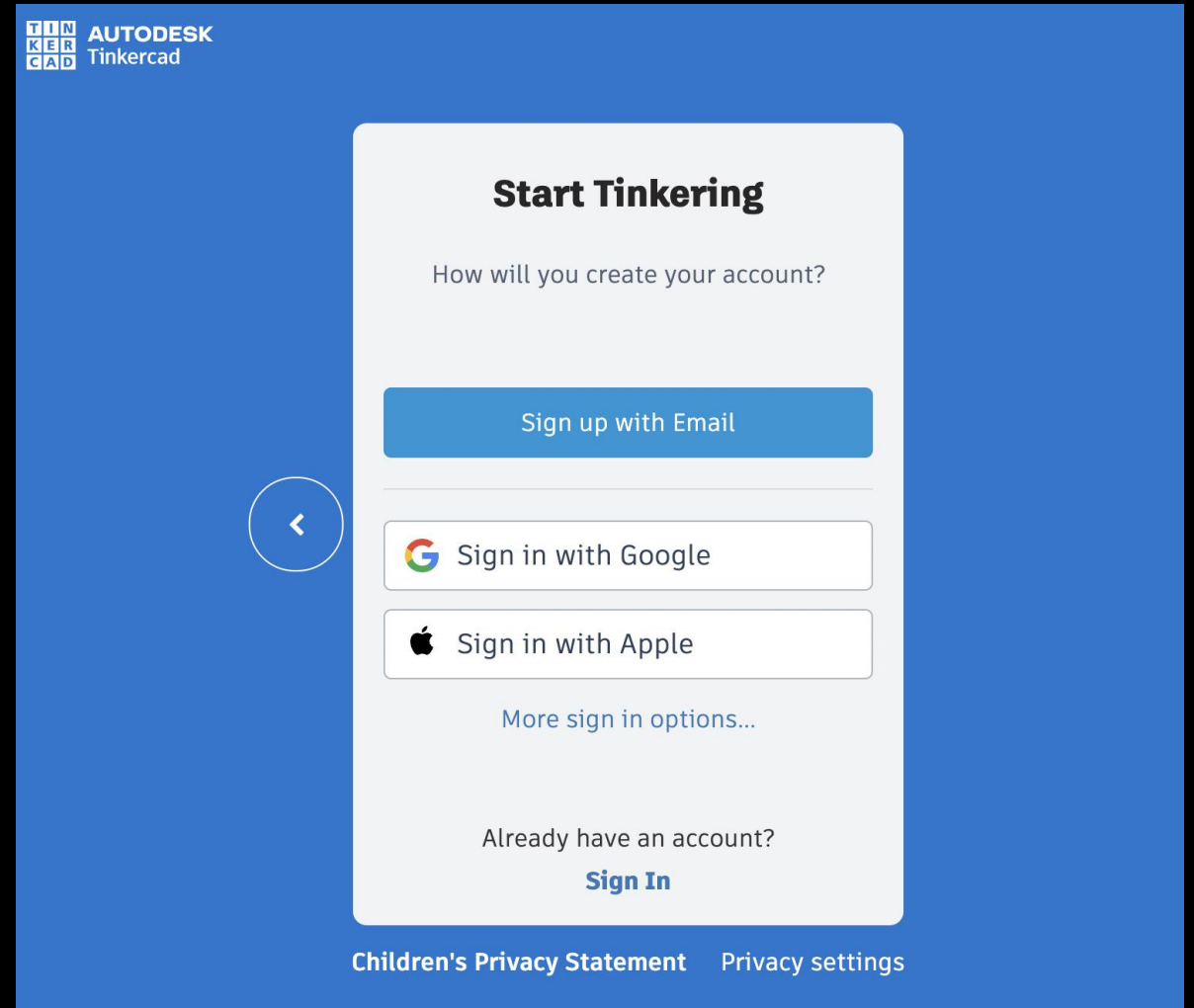
Cons

- Experienced designers might get frustrated with the lack of features

TinkerCAD

Let's have a go!

- Head to the internet and navigate to TinkerCAD.com
- Create a free account using an email address



The Interface

The image shows a 3D CAD software interface with several annotations:

- Auto-generates a file name double click to type in something meaningful**: Points to the file name "Grand Inari" in the top-left corner.
- STEP 1 Get to know your Onscreen Nav tools**: A blue-bordered box pointing to a vertical toolbar on the left side of the interface.
- Step 2 - Shape menu drag basic shapes onto your work plan to start designing**: An orange-bordered box pointing to the "Basic Shapes" panel on the right side of the interface.
- Default Workplane think of this as the floor of you imaginary design space**: A black-bordered box pointing to the blue grid workplane in the center.
- but you can also use this tool to place a workplane on any face of an object**: A black-bordered box pointing to a tool icon in the "Basic Shapes" panel.

The interface includes a top toolbar with icons for file operations, a central 3D workspace with a blue grid and a brown cylinder object, and a right-hand panel with a "Basic Shapes" menu. A "Cylinder" properties panel is also visible, showing options for "Solid" and "Hole" with numerical values for "Sides", "Bevel", and "Segments".

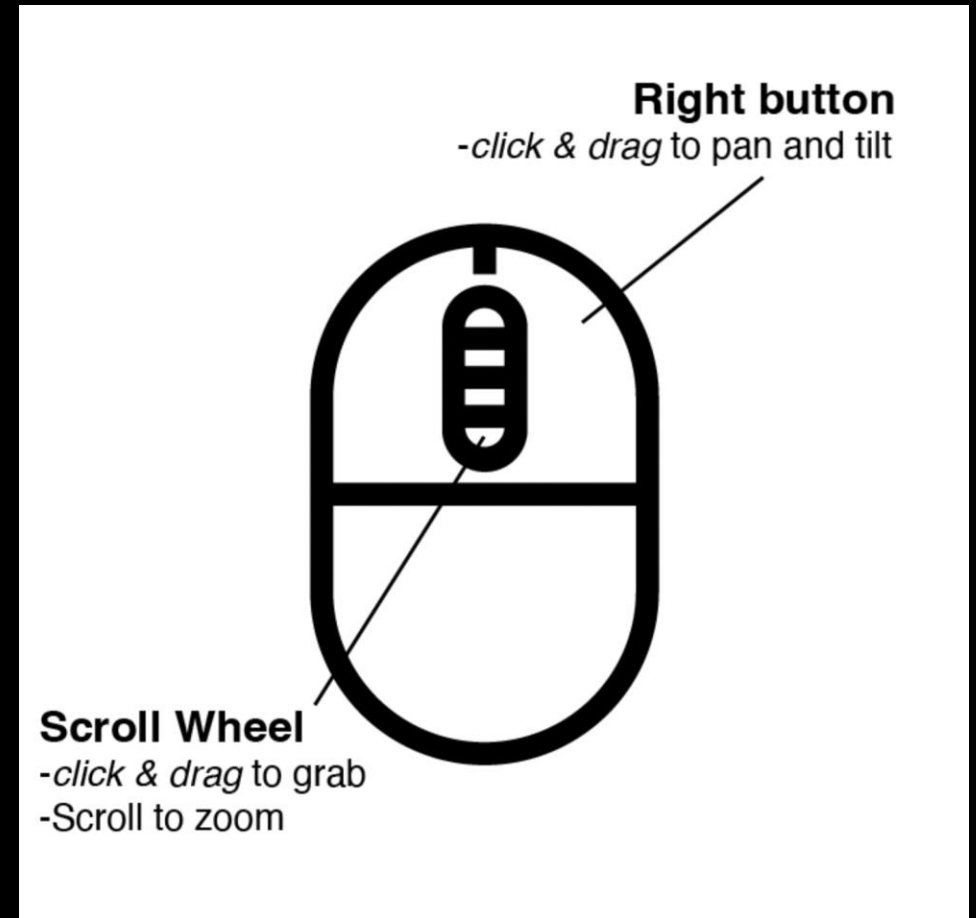
Navigating the 3-dimensional space

Moving around

- Zoom in and out using the scroll wheel

Pan & Tilt

- Right click and drag to change the aspect of your view or
- Click and drag the Cube device to rotate you view angle
- If you get lost click the **home icon** and it will take you back to the default view



Moving and manipulating shapes

Select

- Using the left click (hold shift select multiple objects)
- Or left click and drag a selection box around multiple objects

Resizing

- Left click and drag the black or white handles to resize your shapes
- Hold the SHIFT key to constrain proportions
- Left click and drag the black cone to change the elevation of the shape in relation to the work plane

Moving

- You can nudge a selected object around the x & y axes with the arrow keys

Rotate

- Left click and drag the curly one of the 3 arrows to rotate an object, you can also type in an angle once it is highlighted.

Ruler

- Drag the ruler out onto the workplace to resize or arrange shapes using typed in dimensions

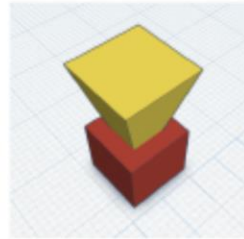
Make a complex shape

Grouping solids and holes

- You can make complex shapes by combining and subtracting the primitive (basic) shapes using the group tool.
- Select the objects you want to combine
- And then hit the group button

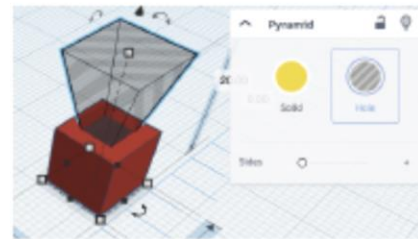
Make a complex 3D shape in 4 steps

1



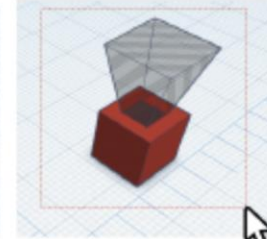
Start with 2 shapes

2



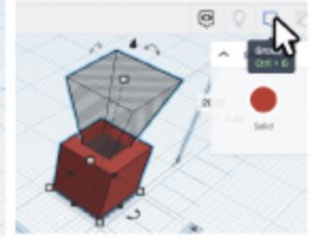
Select one shape and make it a hole

3

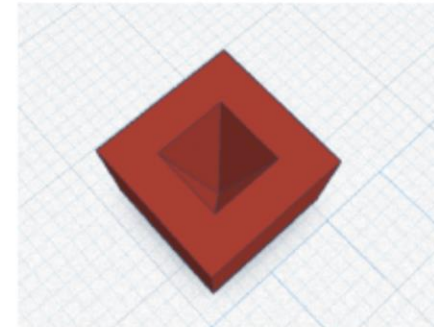


now select both
by dragging a box
around them

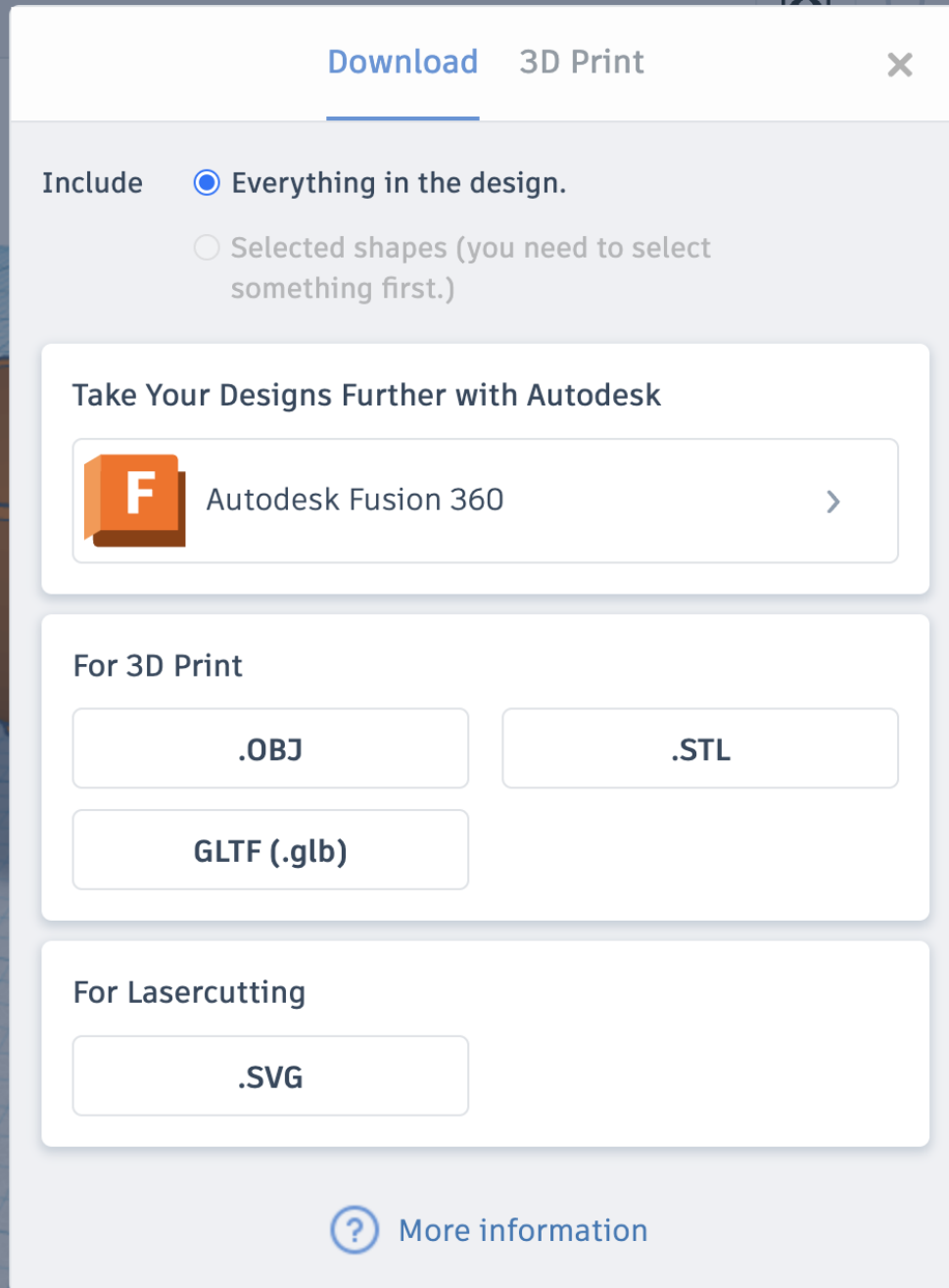
4



and hit the group button



Exporting



- Hit the export button and select STL
- Save to your desired folder
- Open file up in PrusaSlicer using one of our facility computers. The software will be configured correctly on our computers.

What is Slicing?

The act of converting a 3D model into a set of instructions for 3D printers is called **Slicing**. It is the intermediate and most important step in the process of 3D printing and where The Edge staff will need to check your settings.



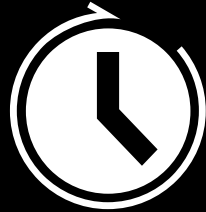
These are some of the main types of settings that can be controlled in a slicer software -

- Print Settings: quality and speed.
- Filament Settings: type of filament etc.
- Printer settings: the model of printer etc.
- Supports: added parts that support overhanging or bridge structures when printing.
- Infill: the internal structure (density/fullness) of a 3D printed part.

Slicing with PrusaSlicer



When choosing your print settings, you need to weigh up some priorities



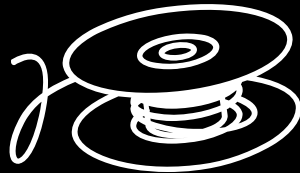
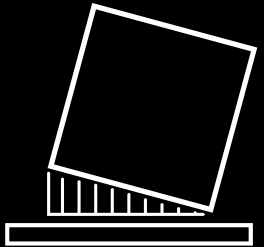
Does it need to be a quick print?

Or do you have time for high quality outcome?



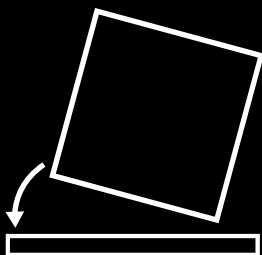
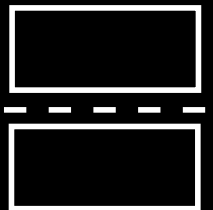
What percentage infill do you need?

Do you need to print support?



How much filament will this use?

Would it be more time or material efficient to cut up your model?



Or maybe reorienting it will save time, plastic, support and or clean up?

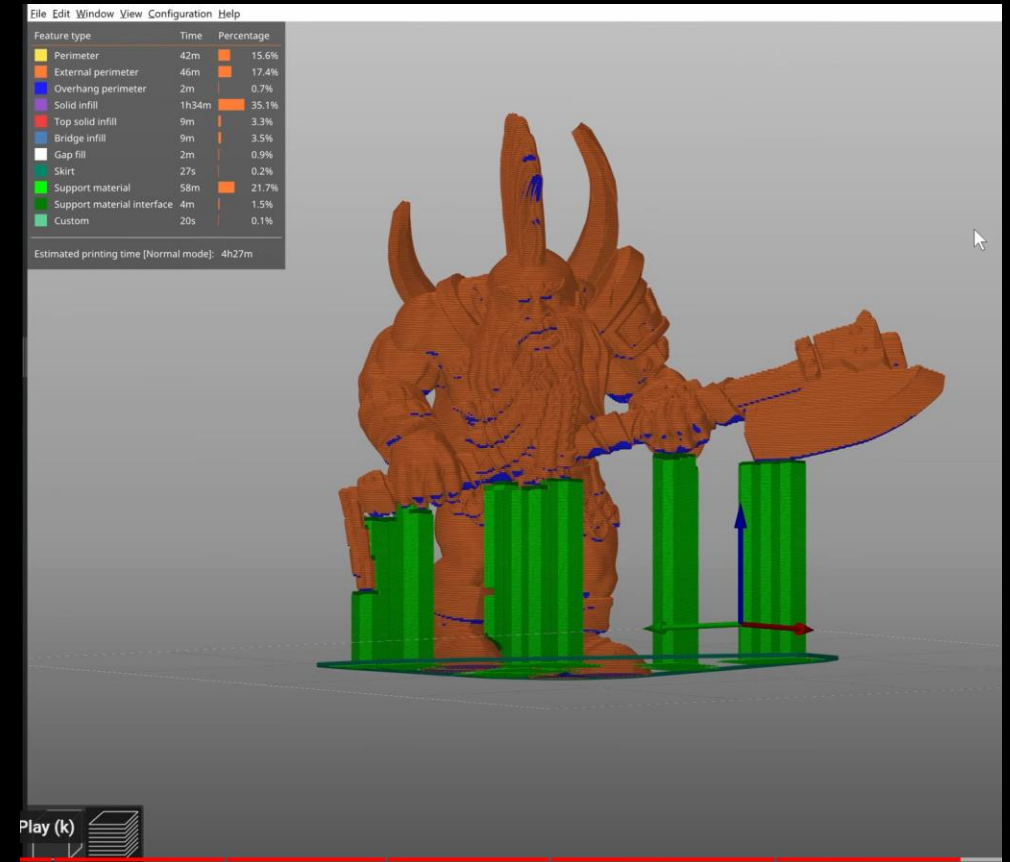
Further info on Supports

3D printers work by layering plastic to create a 3D object. Each new layer must be supported by the one beneath it. If part of your model starts in mid-air and is not supported by anything below, you need to add an additional support structure to ensure a successful print.

Consider if you are able to alter the orientation of the model or split into multiple parts to reduce overhangs and the amount of supports generated.

[PrusaSlicer](#) can both detect areas that need supports and generate them in these places automatically.

The [Paint On support](#) option also works great to manually choose the places you want to add supports to.



[Source](#) Jakub Kočí

Further info on Infill

There are a few factors when choosing an infill pattern, listed below, and you can read up on these more on the [Prusa website](#).

Prints (even mechanical parts) rarely benefit from infill over 20%, as uneven cooling can occur on more complex infill which leads to distortions and delamination.

Print speed: How quickly can the pattern be printed?

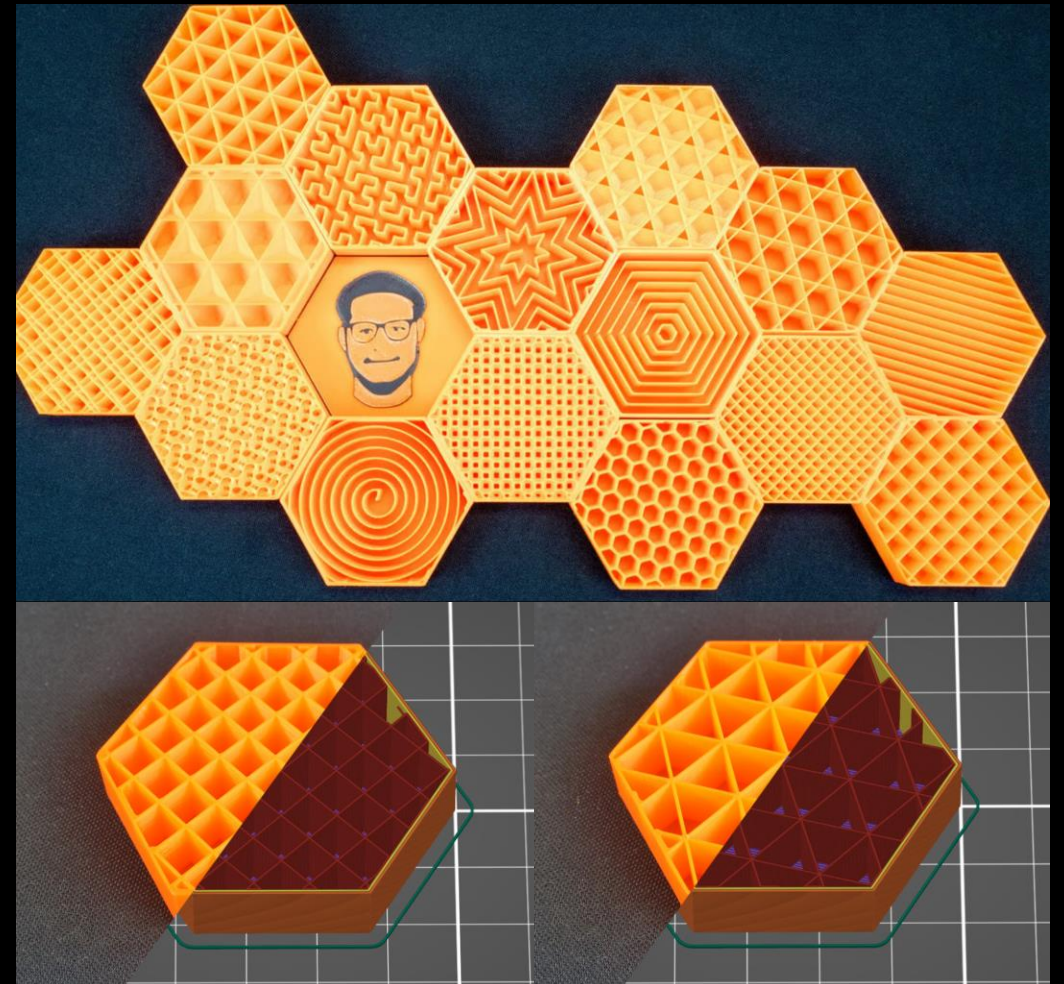
Density: How much filament is used to create the pattern?

Visuals: How does the pattern look when using transparent filament? How does it look if the model is exposed?

Support for top layers: How well will the top layer of a print be supported?

Flexibility: How flexible will the print be with the chosen infill pattern?

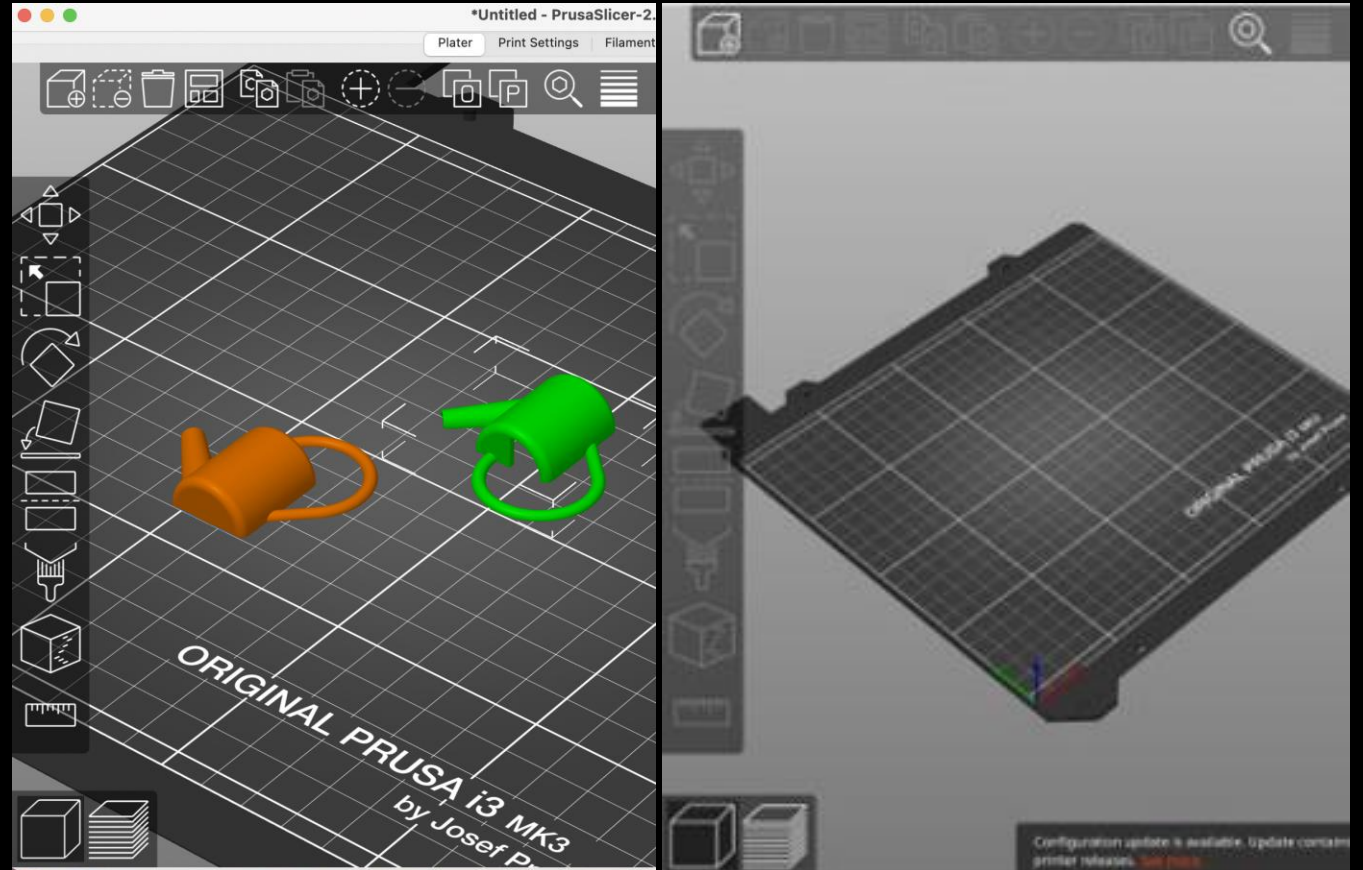
Filling: How suitable is the pattern for filling with resins or other substances? This is especially important for parts that may be used as molds for casting.



[Source](#) [Jakub Kočí](#)

Slicing with PrusaSlicer

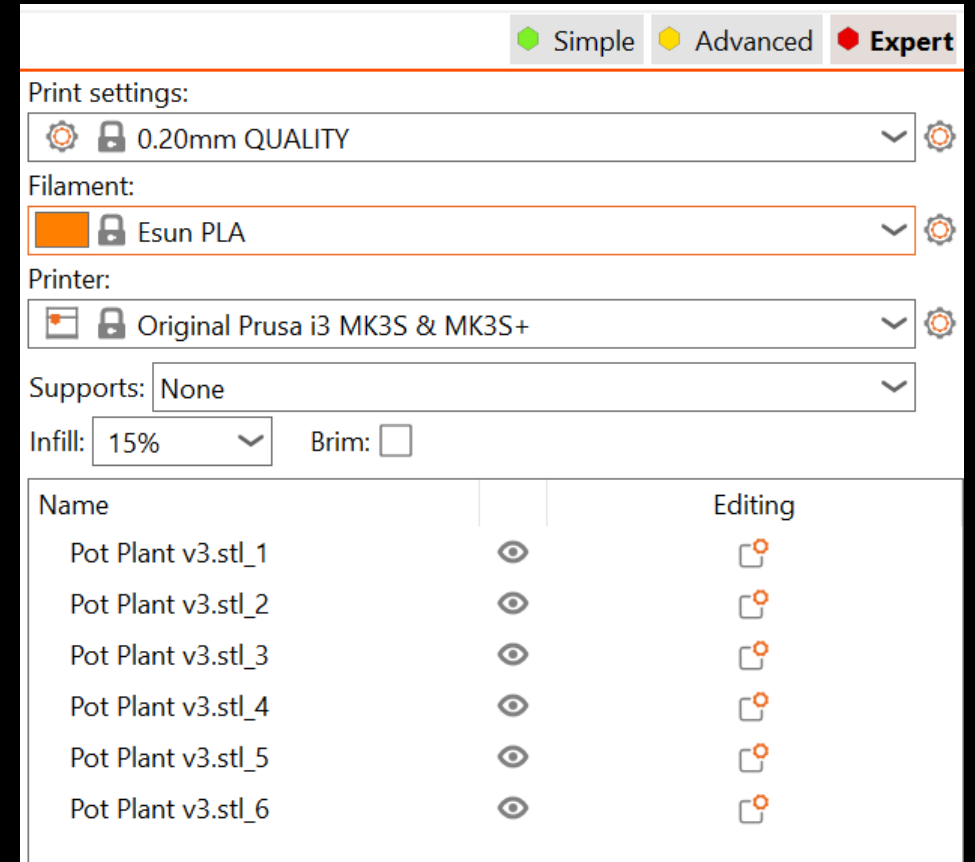
- Open the **PrusaSlicer** application. If you see the pop-up 'Configuration Update' is available, click to install the update before proceeding further.
- Use the **Add** button to import your model/s into PrusaSlicer.
- **Delete /Delete All** buttons to remove models.
- Use **Move, scale, rotate, Place on Face** and **cut tools** to prepare the model to printing in the most efficient/effective orientation.



Slicing with PrusaSlicer

Print Settings

- Select a **Quality / Speed** setting for your print, the higher the quality, the longer the print will take.
- Select the **filament** that corresponds with the one on the printer (check the machine before you slice).
- Select a **Supports** setting
- Select an **Infill** percentage



Slicing with PrusaSlicer

- Click **Slice** to *slice* the model into layers in the Z axis and select **layers preview** to view infill land
- Check the **Sliced Info** for the cost (amount of filament used) and how long it's going to take to print.
- If you are happy with the time and settings, ask a staff member to check over and fill out the print form.
- Generate and export *G-Code* to the SD Card by clicking the **Export G-Code** button.

The screenshot displays the PrusaSlicer 2.7.1 interface. At the top, a 'Sliced Info' window shows the following details:

- Size: 44.74 x 39.49 x 10.00
- Volume: 1697.95
- Facets: 40862 (3 shells)
- 30 open edges

The main interface features a 'Legend' window with a table of feature types, their estimated printing times, and the amount of filament used:

Feature type	Time	Percentage	Used filament
Perimeter	8m	16.7%	0.48 m 1.42 g
External perimeter	14m	29.6%	0.52 m 1.56 g
Overhang perimeter	38s	1.3%	0.02 m 0.07 g
Solid infill	5m	10.8%	0.33 m 0.98 g
Top solid infill	26s	0.9%	0.01 m 0.04 g
Bridge infill	1m	2.3%	0.03 m 0.10 g
Skirt/Brim	25s	0.9%	0.02 m 0.06 g
Support material	10m	21.8%	0.46 m 1.38 g
Support material interface	2m	4.7%	0.08 m 0.23 g
Custom	6s	0.2%	0.02 m 0.06 g

Estimated printing times are shown as: First layer: 4m, Total: 47m.

The right-hand panel shows the 'Print settings' for a 0.15mm QUALITY print using Esun PLA filament on an Original Prusa i3 MK3S & MK3S+ printer. The infill is set to 15% and the brim is disabled. The 'Sliced Info' window at the bottom right repeats the model's dimensions and volume.

The 'Export G-code' button is visible at the bottom right of the interface.

Setting up the printer

STEP BY STEP

After a staff member has approved your slicing, you can head to the 3D printer you have booked to start your 3D print.

1. Turn on the 3D printer (switch located on the back of the Prusa machines)
2. Ensure the printer is set up with the correct filament and build plate.
3. Switch on the extractor under the 3D printer. Each extractor serves 2 printer enclosures. Make sure BOTH the extraction and return air blast gates serving the printer you are using are open. If there is no job on the 2nd printer, ensure the blast gates serving its enclosure are closed.
4. Insert your media (SD card for Mk3s+ or USB thumb drive for the Mk4) into the printer.
5. Navigate to the file on your media by turning the knob on the screen and selecting the correct G-Code file you have prepared. Select by pushing the button. The printer extruder and bed will start heating up. Your print will start when the target temperatures are attained.
6. Monitor the start of the print to make sure it begins with no issues; you can then leave it to print.

Common print failures

- Birds nest (filament non-adhesion)
- X and/or Y-axis shift (belt or stepper motor errors)
- Print wholly or partially detaches from build plate
- Blocked nozzle
- G-code encoding error
- Thermal anomaly or runaway (hot end or bed)
- Head impact

Troubleshooting

Something is weird or going wrong?

- First reaction should always be *pause the job*
- Remember you can always ask a staff member for help!

The filament spool doesn't unwind?

- Check spool mounting and rotation
- Ensure filament on spool is able to unwind without binding

The machine announces an error?

- Refer to the manual for error code explanations
- Check for temperature, connection, or SD card issues



Troubleshooting

The print wholly or partially detaches from the build plate?

- Ensure correct build plate for specific filament type is being used.
- Ensure a clean and level build plate
- Adjust first layer Z height

The printer shifts in the x and/or Y axis?

- Tighten belts and pulleys (staff only)
- Ensure a stable, clear printing environment
- Verify correct print file generation

Filament birds nest forms

- Clean clogged nozzle (staff only)
- Ensure smooth filament path
- Calibrate extruder and temperature settings (staff only)

The spool runs out of filament or the filament breaks?

- Monitor spool and replace before empty (staff only)
- Replace filament with another spool known to be dry (staff only)

See staff for support with any issues labelled Staff only

Extra resources

Prusa

<https://www.prusa3d.com/>

<https://blog.prusa3d.com/>

Thingiverse

Checkout things other people have designed for 3D printing and save yourself a lot of time designing something someone else has successfully made. Remix / customise other people's designs.

<https://www.thingiverse.com/categories>

Functional Print

Check out the functional solutions people print up on Reddit.

<https://www.reddit.com/r/functionalprint/>

THANKS FOR ATTENDING

Please complete our survey that will be sent out via Eventbrite.

Contact us on appliedcreativity@slq.qld.gov.au



slq.qld.gov.au